
Transient analysis

11

Chapter overview

This chapter describes how to set up a transient analysis and includes the following sections:

- [Overview of transient analysis on page 11-360](#)
- [Defining a time-based stimulus on page 11-362](#)
- [Transient \(time\) response on page 11-373](#)
- [Internal time steps in transient analyses on page 11-376](#)
- [Switching circuits in transient analyses on page 11-377](#)
- [Plotting hysteresis curves on page 11-377](#)
- [Fourier components on page 11-379](#)

Overview of transient analysis

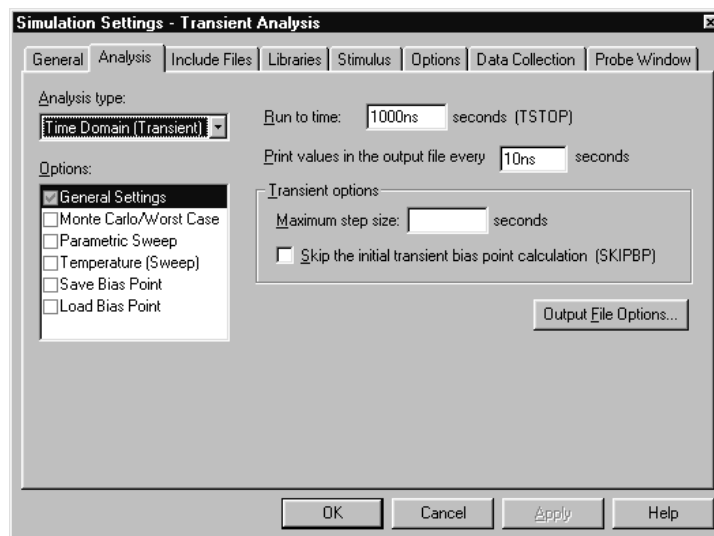
Minimum requirements to run a transient analysis

Minimum circuit design requirements

Circuit should contain one of the following:

- An independent source with a transient specification (see [Table 32](#))
- An initial condition on a reactive element
- A controlled source that is a function of time

Minimum program setup requirements



See [Setting up analyses on page 8-284](#) for a description of the Analysis Setup dialog box.

- 1 From the PSpice menu, choose New Simulation Profile or Edit Simulation Settings. (If this is a new simulation, enter the name of the profile and click OK.)

The Simulation Settings dialog box appears.

- 2 From the Analysis type list box, select Time Domain (Transient).
- 3 Specify the required parameters for the transient analysis you want to run.
- 4 Click OK to save the simulation profile.
- 5 From the PSpice menu, choose Run to start the simulation.

Defining a time-based stimulus

Overview of stimulus generation

Symbols that generate input signals for your circuit can be divided into two categories:

- those whose transient behavior is characterized graphically using the Stimulus Editor
- those whose transient behavior is characterized by manually defining their properties within Capture

Their symbols are summarized in [Table 32](#).

Table 32 Stimulus symbols for time-based input signals

Specified by...	Symbol name	Description
Using the Stimulus Editor	VSTIM	voltage source
	ISTIM	current source
	DIGSTIM1	digital stimuli
	DIGSTIM2	
	DIGSTIM4	
	DIGSTIM8	
	DIGSTIM16	
	DIGSTIM32	
Defining symbol attribute	VSRC	voltage sources
	VEXP	
	VPULSE	
	VPWL	
	VPWL_RE_FOREVER	
	VPWL_F_RE_FOREVER	
	VPWL_N_TIMES	
	VPWL_F_N_TIMES	
	VSFFM	
	VSIN	



Note Stimulus Editor is not included with PSpice A/D Basics.



Note Digital stimuli are not supported in PSpice.

Table 32 *Stimulus symbols for time-based input signals*

Specified by...	Symbol name	Description
	ISRC	current sources
	IEXP	
	IPULSE	
	IPWL	
	IPWL_RE_FOREVER	
	IPWL_F_RE_FOREVER	
	IPWL_N_TIMES	
	IPWL_F_N_TIMES	
	ISFFM	
	ISIN	
	DIGCLOCK	digital clock signal
	STIM1	digital stimuli
	STIM4	
	STIM8	
	STIM16	
	FILESTIM1	digital file stimuli
	FILESTIM2	
	FILESTIM4	
	FILESTIM8	
	FILESTIM16	
	FILESTIM32	



Note *Digital stimuli are not supported in PSpice.*

To use any of these source types, you must place the symbol in your schematic and then define its transient behavior.

Each property-characterized stimulus has a distinct set of attributes depending upon the kind of transient behavior it represents. For VPWL_F_xxx, IPWL_F_xxx, and FSTIM, a separate file contains the stimulus specification.

As an alternative, the Stimulus Editor utility automates the process of defining the transient behavior of stimulus devices. The Stimulus Editor allows you to create analog stimuli which generate sine wave, repeating pulse, exponential pulse, single-frequency FM, and piecewise linear waveforms. It also facilitates creating digital stimuli with complex timing relations. This applies to both stimulus symbols placed in your schematic as well as new ones that you might create.

For information on digital stimuli characterized by property, see [Chapter 14, Digital simulation](#).

The stimulus specification created using the Stimulus Editor is saved to a file, automatically configured into the schematic, and associated with the corresponding VSTIM, ISTIM, or DIGSTIM part instance or symbol definition.

The Stimulus Editor utility



Orcad program versions without the Stimulus Editor must use the characterized-by-property sources listed in [Table 32 on page 11-362](#).

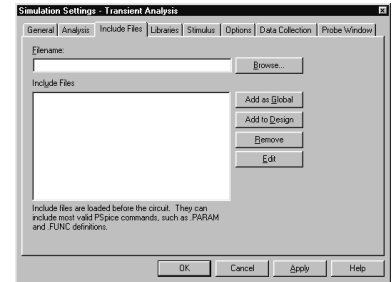
The Stimulus Editor is a utility that allows you to quickly set up and verify the input waveforms for a transient analysis. You can create and edit voltage sources, current sources, and digital stimuli for your circuit. Menu prompts guide you to provide the necessary parameters, such as the rise time, fall time, and period of an analog repeating pulse, or the complex timing relations with repeating segments of a digital stimulus. Graphical feedback allows you to quickly verify the waveform.

Stimulus files

The Stimulus Editor produces a file containing the stimuli with their transient specification. These stimuli are defined as simulator device declarations using the V (voltage source), I (current source), and U STIM (digital stimulus generator) forms. Since the Stimulus Editor produces these statements automatically, you will never have to be concerned with their syntax. However, if you are interested in a detailed description of their syntax, see the descriptions of V and I devices in the *Analog Devices* chapter and stimulus generator in the *Digital Devices* chapter of the the online *PSpice Reference Guide*.

Configuring stimulus files

The Include Files tab in the Simulation Settings dialog box allows you to view the list of stimulus files pertaining to your current schematic. You can also manually add, delete, or change the stimulus file configuration in this tab dialog box. The list box displays all of the currently configured stimulus files. One file is specified per line. Files can be configured as either global to the Capture environment or local to the current design. Global files are marked with an asterisk (*) after the file name.



When starting the Stimulus Editor from Capture, stimulus files are automatically configured (added to the list) as local to the current design. Otherwise, new stimulus files can be added to the list by entering the file name in the Filename text box and then clicking the Add to design (local configuration) or Add as global (global configuration) button.

Starting the Stimulus Editor

The Stimulus Editor is fully integrated with Capture and can be run from either the schematic editor or symbol editor.

You can start the Stimulus Editor by doing the following:

- 1 Select one or more stimulus instances in the schematic
- 2 From the Edit menu, choose PSpice Stimulus.

When you first start the Stimulus Editor, you may need to adjust the scale settings to fit the trace you are going to add. You can use Axis Settings on the Plot menu or the corresponding toolbar button to change the displayed data, the extent of the scrolling region, and the minimum resolution for each of the axes. Displayed Data Range parameters determine what portion of the stimulus data set will be presented on the screen. Extent of Scrolling Region parameters set the absolute limits on the viewable range. Minimum Resolution parameters determine the

smallest usable increment (example: if it is set to 1 msec, then you cannot add a data point at 1.5 msec).

See [Chapter 14, Digital simulation](#), for detailed information about creating digital stimuli.

Defining stimuli

- 1 Place stimulus part instances from the symbol set: VSTIM, ISTIM and DIGSTIMn (found in the SOURCESTM.OLB part library).
- 2 Click the source instance to select it.
- 3 From the Edit menu, choose PSpice Stimulus to start the Stimulus Editor.
- 4 Fill in the transient specification according to the dialogs and prompts.

Piecewise linear and digital stimuli can be specified by direct manipulation of the input waveform display.

- 5 From the File menu, choose Save to save the edits.

Example: piecewise linear stimulus

- 1 Open an existing schematic or start a new one.
- 2 From the Place menu, choose Part and browse the SOURCESTM.OLB part library file for VSTIM (and select it).
- 3 Place the part. It looks like a regular voltage source with an implementation property displayed.
- 4 Click the implementation label and type `vfirst`. This names the stimulus that you are going to create.
- 5 If you are working in a new schematic, use Save from the File menu to save it. This is necessary since the schematic name is used to create the default stimulus file name.
- 6 Click the VSTIM part to select it.
- 7 From the Edit menu, choose PSpice Stimulus. This starts the Stimulus Editor and displays the New

Stimulus dialog box. You can see that the stimulus already has the name of Vfirst.

- 8 Select PWL in the dialog box and click OK. The cursor looks like a pencil. The message in the status bar at the bottom of the screen lets you know that you are in the process of adding new data points to the stimulus. The left end of the bottom status bar displays the current coordinates of the cursor.
- 9 Move the cursor to (200ns, 1) and click the left mouse button. This adds the point. Notice that there is automatically a point at (0,0). Ignore it for now and continue to add a couple more points to the right of the current one.
- 10 Click-right to stop adding points.
- 11 From the File menu, choose Save.

If you make a mistake or want to make any changes, reshape the trace by dragging any of the handles to a new location. The dragged handle cannot pass any other defined data point.

To delete a point, click its handle and press **[Del]**.

To add additional points, either choose Add Point from the Edit menu, press **[Alt]+[A]**, or click the Add Point toolbar button.

At this point you can return to Capture, edit the current stimulus, or go on to create another.

Example: sine wave sweep

- 1 Open an existing schematic or start a new one.
- 2 Place a VSTIM part on your schematic.
- 3 To name the stimulus, double-click the implementation property and type `Vsin`.
- 4 Click the VSTIM part to select it.
- 5 From the PSpice menu, choose Edit Stimulus to start the Stimulus Editor.

This example creates a 10 k sine wave with the amplitude parameterized so that it can be swept during a simulation.

- 6 Define the stimulus parameter for amplitude:
 - a From the New Stimulus dialog box, choose Cancel.
 - b From the Tools menu, choose Parameters.
 - c Enter $AMP=1$ in the Definition text box, and click OK.
 - d From the Stimulus menu, choose New or click the New Stimulus button in the toolbar.
 - e Give the stimulus the name of Vsin.
 - f Select SIN as the type of stimulus to be created, and click OK.
- 7 Define the other stimulus properties:
 - a Enter 0 for Offset Value.
 - b Enter {AMP} for Amplitude. The curly braces are required. They indicate that the expression needs to be evaluated at simulation time.
 - c Enter 10k for Frequency and click OK.
 - d From the File menu, choose Save.
- 8 Within Capture, place and define the PARAM symbol:
 - a From the Place menu, choose Part.
 - b Either browse SPECIAL.OLB for the PARAM part or type in the name.
 - c Place the part on your schematic and double-click it.
 - d Click New to add a new user property.
 - e Set the value property name to AMP (no curly braces).
 - f Set the value of the AMP property to 1.
- 9 Set up the parametric sweep and other analyses:
 - a From the PSpice menu, choose Edit Simulation Profile, and click the Parametric Sweep button.
 - b Select Global Parameter in the Swept Var. Type frame.

- c Select Linear in the Sweep type frame.
- 10 Enter AMP in the Name text box.
- 11 Specify values for the Start Value, End Value, and Increment text boxes.

You can now set up your usual Transient, AC, or DC analysis and run the simulation.

Creating new stimulus symbols

- 1 Use the Capture part editor to edit or create a part with the following properties:

Implementation Type	PSpice Stimulus
Implementation	name of the stimulus model
STIMTYPE	type of stimulus; valid values are ANALOG or DIGITAL; if this property is nonexistent, the stimulus is assumed to be ANALOG

Editing a stimulus

To edit an existing stimulus

- 1 Start the Stimulus Editor and select Get from the Stimulus menu.
- 2 Double-click the trace name (at the bottom of the X axis for analog and to the left of the Y axis for digital traces.) This opens the Stimulus Attributes dialog box where you can modify the attributes of the stimulus directly and immediately see the effect of the changes.

PWL stimuli are a little different since they are a series of time/value pairs.

To edit a PWL stimulus

- 1 Double click the trace name. This displays the handles for each defined data point.
- 2 Click any handle to select it. To reshape the trace, drag it to a new location. To delete the data point, press **Del**.
- 3 To add additional data points, either select Add from the Edit menu or click the Add Point button.
- 4 Right-click to end adding new points.

This provides a fast way to scale a PWL stimulus.

To select a time and value scale factor for PWL stimuli

- 1 Select the PWL trace by clicking on its name.
- 2 Select Attributes from the Edit menu or click the corresponding toolbar button.

Deleting and removing traces

To delete a trace from the displayed screen, select the trace name by clicking on its name, then press **[Del]**. This will only erase the display of the trace, not delete it from your file. The trace is still available by selecting Get from the Stimulus menu.

To remove a trace from a file, select Remove from the Stimulus menu.

Note *Once a trace is removed, it is no longer retrievable. Delete traces with caution.*

Manual stimulus configuration

Stimuli can be characterized by manually starting the Stimulus Editor and saving their specifications to a file. These stimulus specifications can then be associated to stimulus instances in your schematic or to stimulus symbols in the symbol library.

To manually configure a stimulus

- 1 Start the Stimulus Editor by double-clicking on the Stimulus Editor icon in the Orcad program group.
- 2 Open a stimulus file by selecting Open from the File menu. If the file is not found in your current library search path, you are prompted for a new file name.
- 3 Create one or more stimuli to be used in your schematic. For each stimulus:
 - a Name it whatever you want. This name will be used to associate the stimulus specification to the stimulus instance in your schematic, or to the symbol in the symbol library.
 - b Provide the transient specification.
 - c From the File menu, choose Save.

- 4 In the schematic page editor, configure the Stimulus Editor's output file into your schematic:
 - a From the Pspice menu, choose Edit Simulation Profile.
 - a In the Simulation Settings dialog box, select the Include Files tab.
 - b Enter the file name specified in step 2.
 - c If the stimulus specifications are for local use in the current design, click the Add to design button. For global use by any design, use Add as global instead.
 - d Click OK.
- 5 Modify either the stimulus instances in the schematic or symbols in the symbol library to reference the new stimulus specification.
- 6 Associate the transient stimulus specification to a stimulus instance:
 - a Place a stimulus part in your schematic from the part set: VSTIM, ISTIM, and DIGSTIMn.
 - b Click the VSTIM, ISTIM, or DIGSTIMn instance.
 - c From the Edit menu, choose Properties.
 - d Click the Implementation cell, type in the name of the stimulus, and click Apply.
 - e Complete specification of any VSTIM or ISTIM instances by selecting Properties from the Edit menu and editing their DC and AC attributes.

Click the DC cell and type its value.

Click the AC cell, type its value, and then click Apply.
 - f Close the property editor spreadsheet.

- 7 To change stimulus references globally for a part:
 - a Select the part you want to edit.
 - a From the Edit menu, choose Part to start the part editor.
 - b Create or change the part definition, making sure to define the following properties:

Implementation	stimulus name as defined in the Stimulus Editor
----------------	---

See [Chapter 5, Creating parts for models](#), for a description of how to create and edit parts.

Transient (time) response

The Transient response analysis causes the response of the circuit to be calculated from $\text{TIME} = 0$ to a specified time. A transient analysis specification is shown for the circuit EXAMPLE.OPJ in Figure 64. (EXAMPLE.OPJ is shown in Figure 65.)

The analysis is to span the time interval from 0 to 1000 nanoseconds and values should be reported to the simulation output file every 20 nanoseconds.

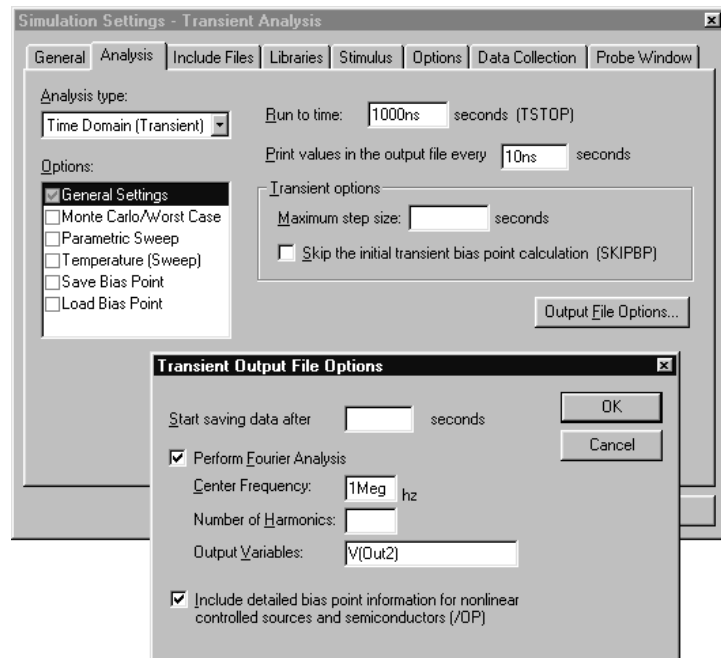
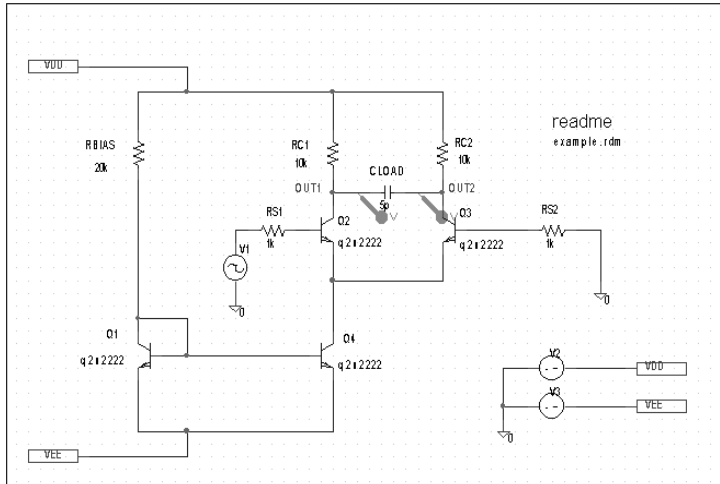


Figure 64 Transient analysis setup for EXAMPLE.OPJ.

During a transient analysis, any or all of the independent sources may have time-varying values. In EXAMPLE.OPJ, the only source which has a time-varying value is V1 (VSIN part) with attributes:

$$\begin{aligned} V_{OFF} &= 0\text{v} \\ V_{AMPL} &= 0.1\text{v} \\ FREQ &= 5\text{Meg} \end{aligned}$$

V1's value varies as a 5 MHz sine wave with an offset voltage of 0 volts and a peak amplitude of 0.1 volts. In general, more than one source has time-varying values; for instance, two or more clocks in a digital circuit.



The example circuit EXAMPLE.OPJ is provided with the Orcad program installation.

Figure 65 Example schematic EXAMPLE.OPJ.

The transient analysis does its own calculation of a bias point to start with, using the same technique as described for DC sweep. This is necessary because the initial values of the sources can be different from their DC values. To report the small-signal parameters for the transient bias point, use the Transient command and enable Detailed Bias Point. Otherwise, if you simply want the result of the transient run itself, you should only enable the Transient command.

In the simulation output file EXAMPLE.OUT, the bias-point report for the transient bias point is labeled INITIAL TRANSIENT SOLUTION.

Internal time steps in transient analyses

During analog analysis, PSpice maintains an internal time step which is continuously adjusted to maintain accuracy while not performing unnecessary steps. During periods of inactivity, the internal time step is increased. During active regions, it is decreased. The maximum internal step size can be controlled by specifying it in the Maximum Time Step text box in the Transient dialog box. PSpice will never exceed either the step ceiling value or two percent of the total transient run time, whichever is less.

The internal time steps used may not correspond to the time steps at which information has been requested to be reported. The values at the print time steps are obtained by second-order polynomial interpolation from values at the internal steps.

See [Chapter 14, Digital simulation](#), for more information on the digital timing analysis of PSpice A/D.

When simulating mixed analog/digital circuits, there are actually two time steps: one analog and one digital. This is necessary for efficiency. Since the analog and digital circuitry usually have very different time constants, any attempt to lock them together would greatly slow down the simulation. The time step shown on the PSpice display during a transient analysis is that of the analog section.

Switching circuits in transient analyses

Running transient analysis on switching circuits can lead to long run times. PSpice must keep the internal time step short compared to the switching period, but the circuit's response extends over many switching cycles.

One method of avoiding this problem is to transform the switching circuit into an equivalent circuit without switching. The equivalent circuit represents a sort of quasi steady-state of the actual circuit and can correctly model the actual circuit's response as long as the inputs do not change too fast.

This technique is described in:

V. Bello, "Computer Program Adds SPICE to Switching-Regulator Analysis," *Electronic Design*, March 5, 1981.

Plotting hysteresis curves

Transient analysis can be used to look at a circuit's hysteresis. Consider, for instance, the circuit shown in Figure 66 (netlist in Figure 67).

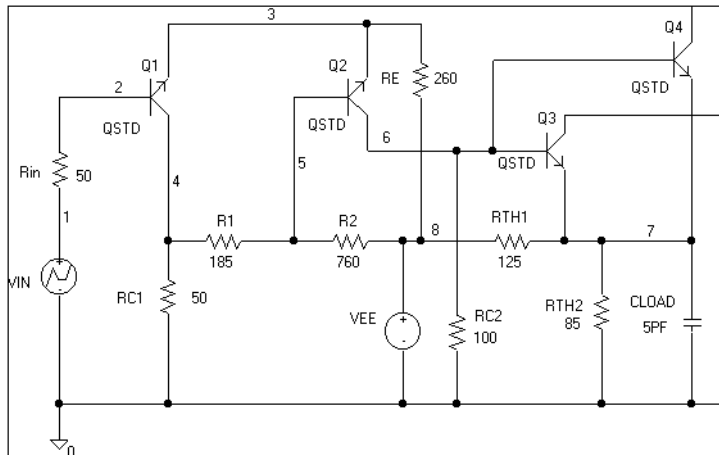


Figure 66 ECL-compatible Schmitt trigger.

```

* Capture Netlist

R_RIN      1 2 50
R_RC1      0 3 50
R_R1       3 5 185
R_R2       5 8 760
R_RC2      0 6 100
R_RE       4 8 260
R_RTH2     7 0 85
C_CLOAD    0 7 5PF
V_VEE      8 0 dc -5
V_VIN      1 0
+PWL 0 -8 1MS -1.0V 2MS -1.8V
R_RTH1     8 7 125
Q_Q1       3 2 4 QSTD
Q_Q2       6 5 4 QSTD
Q_Q3       0 6 7 QSTD
Q_Q4       0 6 7 QSTD

```

Figure 67 Netlist for Schmitt trigger circuit.

The QSTD model is defined as:

```

.MODEL QSTD NPN( is=1e-16 bf=50 br=0.1 rb=50 rc=10
tf=.12ns tr=5ns
+ cje=.4pF pe=.8 me=.4 cjc=.5pF pc=.8 mc=.333 ccs=1pF
va=50)

```

Instead of using the DC sweep to look at the hysteresis, use the transient analysis, (Print Step = .01ms and Final Time = 2ms) sweeping VIN from -1.8 volts to -1.0 volts and back down to -1.8 volts, very slowly. This has two advantages:

- it avoids convergence problems
- it covers both the upward and downward transitions in one analysis

After the simulation, in the Probe window in PSpice, the X axis variable is initially set to be Time. By selecting X Axis Settings from the Plot menu and clicking on the Axis Variable button, you can set the X axis variable to be V(1). Then use Add on the Trace menu to display V(7), and change the X axis to a user defined data range from -1.8V to -1.0V (Axis Settings on the Plot menu). This plots the output of the Schmitt trigger against its input, which is the desired outcome. The result looks similar to Figure 68.

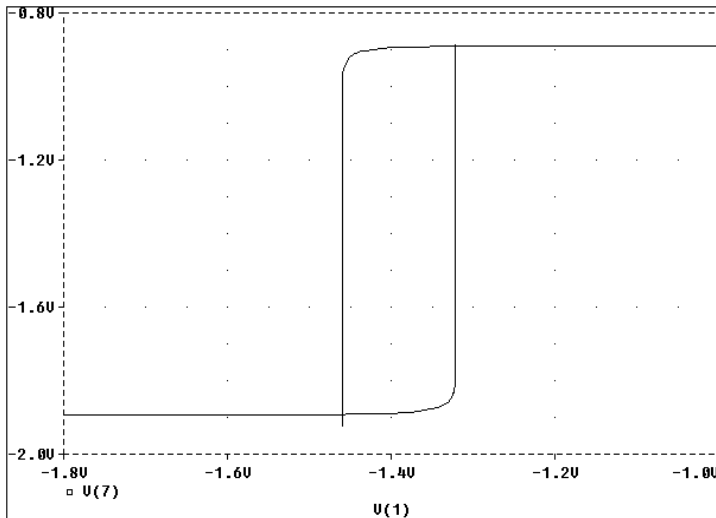


Figure 68 Hysteresis curve example: Schmitt trigger.

Fourier components

Fourier analysis is enabled through the Output File Options dialog box under the Time Domain (Transient) Analysis type. Fourier analysis calculates the DC and Fourier components of the result of a transient analysis. By default, the first through ninth components are computed; however, more can be specified.

When selecting Fourier to run a harmonic decomposition analysis on a transient waveform, only a portion of the waveform is used. Using the Probe window in PSpice, a Fast Fourier Transform (FFT) of the complete waveform can be calculated and its spectrum displayed.

In the example shown in Figure 64 on page 11-374, the voltage waveform at node OUT2 from the transient analysis is to be used and the fundamental frequency is to be one megahertz for the harmonic decomposition. The period of fundamental frequency is one microsecond (inverse of the fundamental frequency). Only the last one microsecond of the transient analysis is used, and that

Note You must do a transient analysis in order to do a Fourier analysis. The sampling interval used during the Fourier transform is equal to the print step specified for the transient analysis.

portion is assumed to repeat indefinitely. Since V1's sine wave does indeed repeat every one microsecond, this is sufficient. In general, however, you must make sure that the fundamental Fourier period fits the waveform in the transient analysis.

Parametric and temperature analysis

12

Chapter overview

This chapter describes how to set up parametric and temperature analyses. Parametric and temperature are both simple multi-run analysis types.

This chapter includes the following sections:

- [Parametric analysis on page 12-382](#)
- [Temperature analysis on page 12-391](#)